



AD-TECH *inc*

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Pro/ENGINEER Interface for CATIA V5 with ATB

Associative data exchange

Industry-wide, dealing with CAD data from a variety of sources is an absolute necessity. Pro/ENGINEER Interface for CATIA V5 with ATB makes that reality a bit easier. It reads and writes CATIA V5 modeling data into and out of Pro/ENGINEER, while retaining associativity (import only). Changes made to the CATIA models drive updates within Pro/ENGINEER, allowing more efficient multi-CAD design.

Capabilities & Benefits

- Enables import of CATIA V5 parts (V5R6 through V5R11, Wildfire 2.0)
- Enables import and export of CATIA V5 parts and assemblies (V5R10 through V5R16, Wildfire 3.0 & 4.0)
- Supports 3D geometry, topology, color, and show/no show attribute transfer
- Native data import without requiring a CATIA V5 license
- Associative updates allow concurrent design and production work in Pro/ENGINEER to proceed with confidence

